Heat Conduction in a Thin, Heated Plate
ME 448/548: StarCCM+ Tutorial
Gerald Recktenwald[*]
18 January 2022

Introduction

Heat conduction in a thin, heated plate is used to introduce StarCCM+, a software system for simulating fluid flow and solid mechanics. A step-by-step procedure is presented for importing a simple CAD model, defining the physical behavior of the material, applying boundary conditions, generating a mesh, controlling the solution and visualizing the results. Screenshots in the tutorial were taken from StarCCM+ version 13.02 (2018). The user interface (UI) components may have a slightly different appearance in later versions.

The tutorial is a recipe of steps, not a comprehensive guide to StarCCM+. This version of the tutorial uses parts-based meshing.

Model Geometry and Boundary Conditions

Figure 1 is a top view of a serpentine heater attached to one side of a relatively thin metal plate. The side of the plate opposite of the heater is cooled by convection. The engineering question of interest is whether the temperature on the convectively cooled surface is sufficiently uniform. For example, suppose that this heater assembly is to be used in a heat transfer experiment that requires a uniform temperature surface exposed to a flowing fluid. The serpentine heater is attached to the back side – away from the fluid. The metal plate provides a structural boundary to contain the flow and the metal helps to spread the heat from the serpentine heater to the surface exposed to the fluid. The thicker the plate, the better the heat spreading. An engineering design decision is how thick to make the fluid side of plate have a surface temperature with a spatial variability less than some tolerance.

Table 1 lists the physical dimensions of the plate and heater. Not all of the dimensions in Figure 1 and Table 1 are independent. The relationships between dimensions are

$$H = h + 2c \ \Rightarrow c = \frac{1}{2}(H - h)$$

$$W = 2s + (n - 1)b + wh \ \Rightarrow s = \frac{1}{2}[W - (n - 1)b - w_h]$$

We assume that the geometry of the plate and heater are defined in a CAD tool such as Solidworks. StarCCM+ has a built-in CAD modeler, so the choice to import the geometry from another software is somewhat arbitrary. The heater is inscribed on the surface of the plate, which means that the thickness of the heater is negligible. In other words, the entire computational model is contained in rectangular sheet of uniform thickness.

---

[*] Mechanical and Materials Engineering Department, Portland State University, gerry@pdx.edu
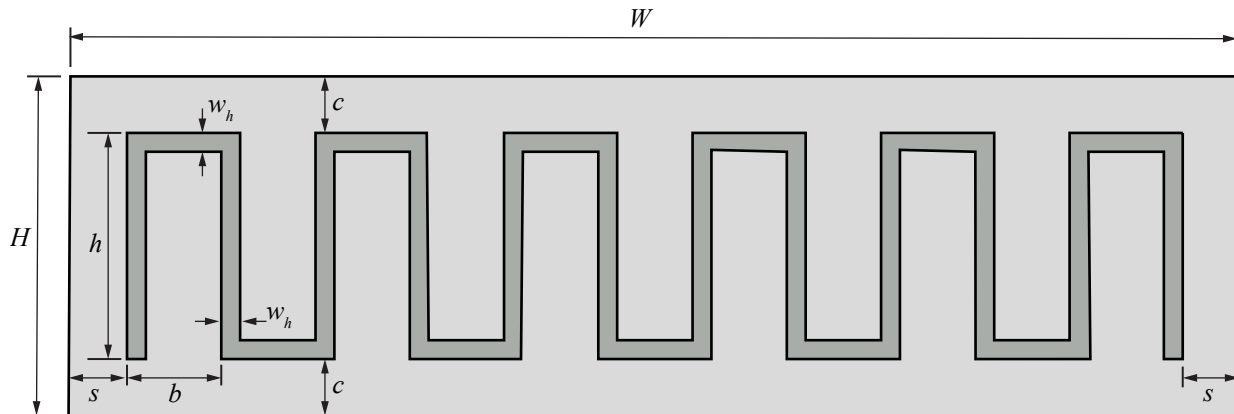
Figure 1   Sketch of the heater strip on the top surface of the heated plate.

Table 1 Dimensions of features in the solid model of the heated plate.

| Symbol	Value	Description
W	10 cm	width of the plate
H	3 cm	height of the plate
t	2 mm	thickness of the plate
wh	2 mm	width of heater strip
h	2 cm	vertical length of heater strip
b	8 mm	horizontal spacing between heater strips
c	5 mm	Vertical distance from heater strips to edge of the plate
s	5 mm	Horizontal distance from heater strips to edge of the plate
n	12	number of heater strips in horizontal direction

Build model in Solidworks

We assume that the reader has sufficient experience with Solidworks to use the following recipe to create the CAD part used in the StarCCM+ model.

- Create the plate by sketching and extruding a rectangle

- Sketch on the top surface to create construction lines.

- Using the construction lines as guide, create a unit of the serpentine pattern that can be replicated. Dimension the unit shape before replicating. Edit the heater shape so that it is a closed contour. This process is tedious.

- Wrap the closed contour of the heater onto the top surface of the plate – use the *Cut* option, not emboss.

- Export as (*Save as…*) a Parasolid model (.x_t).

Import the Parasolid Model into Star

- File → Import → Import Surface Mesh. Select the .x_t file

- Accept defaults, as shown in the screen shot to the right

- The part should be visible in a new *Geometry Scene*.
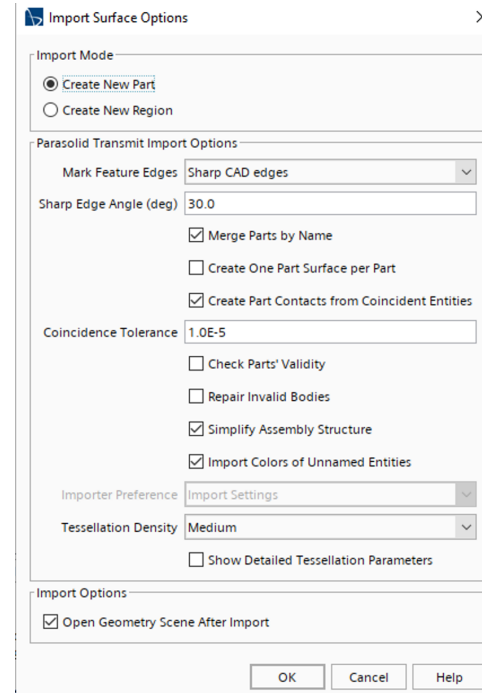
Rename surfaces on the geometric parts

Although it is possible to accept the default names for parts and part surfaces, I strongly recommend that you assign meaningful names to geometric objects at the very start of the simulation. The StarCCM+ interface propagates those names throughout the model-building, simulation, and post-processing phases. Descriptive and unambiguous names for surfaces, regions and other model components will reduce the odds of making a model-building error, and it will make it easier to extract and present results of the StarCCM+ simulation.

Regardless of how you name parts, you will need to separate the boundary surfaces of geometry in order to impose boundary conditions. Thus, the *split-by-patch* step (or some equivalent maneuver) described below is an essential step in model-building.

- Expand the Geometry node and then the Parts node in the simulation tree. Note that the name "heated_plate" is automatically assigned to the part from the name of the Parasolid file you imported.

- Expand the Surfaces node of the heated_plate part.

- Select the Faces node and right click Split by patch. This opens up a new panel (Split Part Surface by Patch) in the space where the simulation tree was located on the left side of the user interface (UI). The panel has a pane with a list of surfaces identified by number. Clicking on one of the numbers highlights the corresponding surface in the geometry scene.

- Select the serpentine heater surface, either in the geometry window, or by number in the list on the left. You may need to rotate the model in the geometry scene.

» With the serpentine heater selected, enter "`heater surface`" in the Part Surface Name box. This name will distinguish the surface with the rest of the top face.

» Click Create at the bottom of the panel. When you create a surface with this step, you are essentially renaming it. The newly created (renamed) surface disappears from the list of numbered surfaces. Note that you can select multiple surfaces, even non-contiguous (non-adjacent) surfaces, and group them as a single named surface.

» After the heater is named in the previous step, it disappears in the scene! As you work your way through naming surfaces, you reduce the visible surfaces, which can help to both identify and select the remaining (unnamed) surfaces.

- Select the remainder of the rectangular face that is coplanar with the serpentine heater.

  » Enter "`top surface`" in the Part Surface Name box

  » Click Create.

- Select the bottom surface

  » Enter "`bottom surface`" in the Part Surface Name box

  » Click Create.

At this point, the geometry scene has four remaining surfaces – the edges of the plate as shown in Figure 2. The four edges will be labeled as a single surface, which allows us (later) to apply the same boundary condition to the edges with one set of parameters. We will rename these from the simulation tree, not in the Split Part Surface by Patch panel.
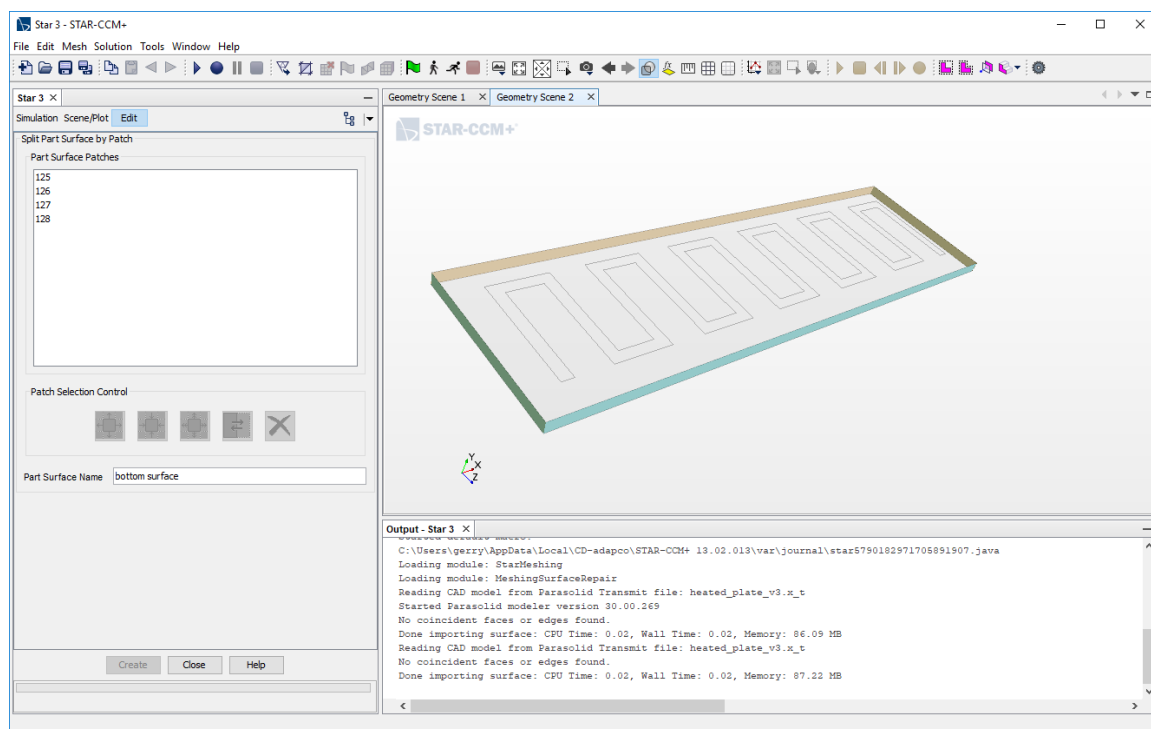

Figure 2   Geometry Scene after all surfaces except the plate edges have been named.

Rename the remaining edges of the plate

- Close the Split Part Surface by Patch panel by clicking Close at the bottom of the panel.

- Notice the list of surfaces in the simulation tree now includes the newly named faces, along with the generic ColoredFace1 surface, which (if you followed preceding steps) should be the edges of the plate. Rename the edges (right click, Rename…) as "plate edges"

- Check your selections by clicking on each of the surfaces under the Surfaces node. Selecting a name should highlight the corresponding surface in the Geometry Scene. If not, you may try to rename the surfaces. You may need to re-import the surface mesh for the geometry and do the split-by-patch and naming operations again.

Save your model

When working with StarCCM+, it is important to run a simulation from a model stored on a local drive. In other words, *don't run your model over the network* where it is saved on the N: drive.
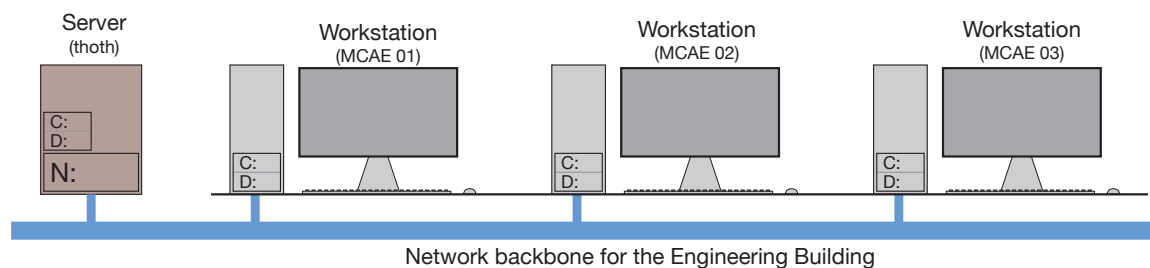

Figure 3   Workstations on the MCECS network

Figure 3 is a schematic of computers on the MCECS network. The blue lines represent the wired network in the Engineering Building. Each of the workstations in the MCAE lab, as well as in offices and other labs in the building, is connected to the network backbone and the Windows file server (thoth). Usually we work with our files on the server, which allows us to move from machine to machine without having to worry about where our files are stored.

When running a simulation, StarCCM+ makes heavy use of disk storage. If you save your model on the N: drive of the server, your StarCCM+ communication with the file system is injected into the building network. This can put a load on the network, causing network slowing, which will impede other users on the network. Working from the N: drive will also cause your simulation to run more slowly, especially if you are using complex models with a large mesh.

To avoid straining the network while running StarCCM+, store your model on the local scratch drive (D:) of the workstation you are using. When your simulation session is over, copy the saved model to your network drive or to a local hard drive or USB drive attached to the computer.

Note: Running StarCCM+ with your simulation files on the D: drive creates a risk that you will forget to move your files back to the N: drive when you are done.

As your simulations get more complex, your models and data may also use up significant storage on the server, and possibly exceed your disk quota. I recommend using an external USB

storage device, such as a large flash drive or an external disk to back up your work. Regardless of the location for your simulation, be sure to save your model in stages as you build it.

Assign Parts to Region

Parts are geometric entities. Regions are domains (or sub-domains) that are assigned physical properties (fluid, solid, equations of state), model physics (laminar, turbulent, compressible, influenced by gravity, isothermal) and meshes (mesh type, resolution), and boundary conditions. A StarCCM+ simulation can have multiple regions and be comprised of multiple parts.

- Select the `heated_plate` part under the Parts sub-node under the Geometry node.

  » Right click and select Assign Parts to Regions

  » Step 1 in the screenshot: select the part by clicking the little blue square in the Assign parts to Regions panel.

  » Step 2 in the screenshot: **IMPORTANT:** Select Create Boundary for Each Surface Part in the bottom part of the panel

  » Step 3 in the screenshot: give the region a name, say "`solid`"

  » Leave other selections to their default values

  » Click Apply and click Close.
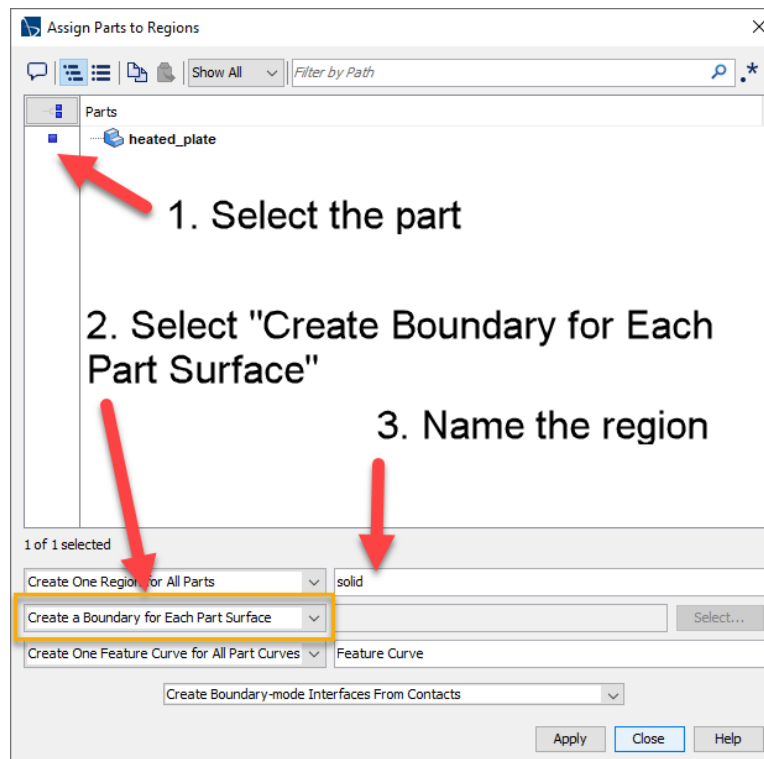
Create the Physics Continua

A *Physics Continua* defines the physical properties of the material. The Physics Continua also specifies behaviors that a region can exhibit, for example whether the material flows or is a solid, whether the flow is laminar or turbulent, or whether heat transfer is present. In complex models, e.g. with both fluid and solid, or with more than one type of fluid, there can be more than one physics continua.

Create the Physics Continua for the Solid

- In the main simulation tree, expand the Continua node. A Physics 1 node may already exist. If not, right-click on Continua, and select New → Physics Continuum.

- Right click on Physics 1 and rename it to "`solid physics`" or just "`solid`"

- Right click on Models → Select Models. In the solid Model Selection modal window and make the following selections. As you make selections, the Model Selection window adapts to indicate enabled models on the right, and new choices on the left.

    » Three Dimensional (may already be selected)

    » Solid

    » Steady

    » Segregated Solid Energy

    » Constant density

    » Click Close.
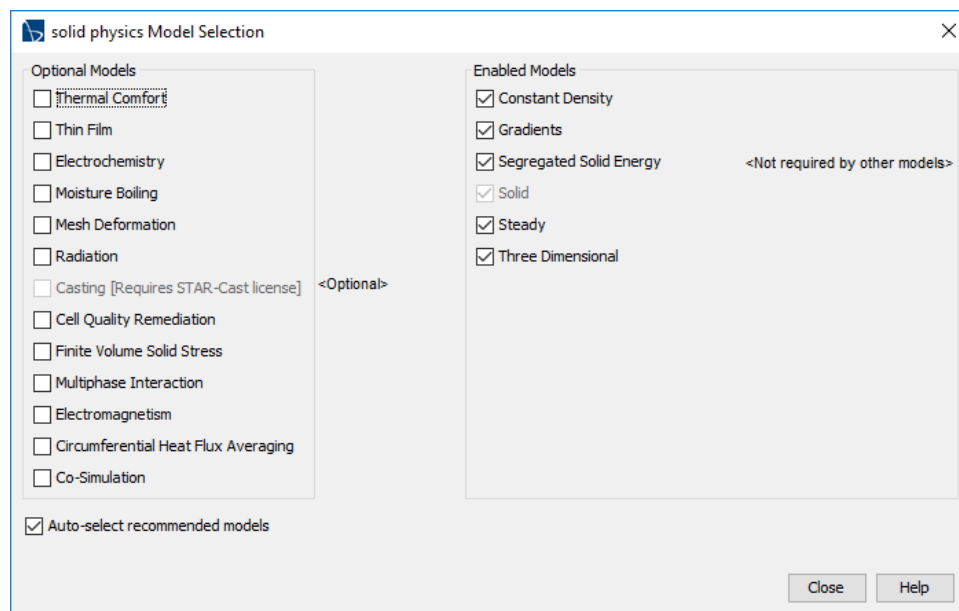
Select the Materials

- Expand the Models sub-node under the (newly created) solid (physics continuum) node.

- Expand the Solid node (click on the + sign)

» Observe that the default material is Aluminum. That will be OK for this first simulation. Note that the material of the serpentine heater is not a factor since we are assuming that the heater has negligible thickness and is inscribed on the surface of the aluminum plate.

- Save the model.

Boundary Conditions

Boundary conditions have a large impact on the behavior of the model. We will choose the following boundary conditions.

- Prescribe a total heat transfer rate on the serpentine heater surface. We could also specify a heat flux, but that would require a manual calculation of the area of the heater in order to specify the heat flux that gives the desired total heat transfer rate.

- The area of the top face of the plate that is in the plane of the heater and that is not covered by the heater, is assumed to be adiabatic. The adiabatic boundary condition is consistent with assuming that the entire face containing the heater is well insulated.

- The narrow edges of the plate are adiabatic.

- The bottom face, opposite from the heater has a convective boundary condition.

This boundary condition specification means that all of the energy added by the heater must leave through the convective surface. We will check the consistency of the model with an energy balance after the simulation is completed.

Apply Boundary Conditions to the Solid Region

- In the simulation tree, expand the Regions node; expand the solid node; expand the Boundaries node.

- Expand the heated_plate.plate plate edges node.

    » Expand the Physics Conditions node.

    » Click on Thermal Specification.

    » Notice that in the Properties pane, the thermal condition is Adiabatic. That is the default, and we want to keep it that way. No modifications are necessary, but it is always good to verify the boundary conditions for each surface.

Note that when you change values in the Properties pane, there is no "OK" or "Close" button. Simply type in the new value and press "return" or "enter" on your keyboard.
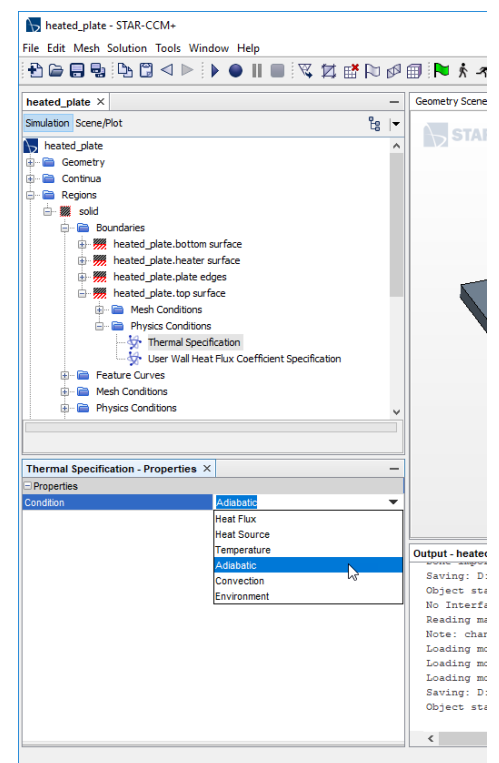
- Under the Boundaries node of the simulation tree, expand the heated_plate.bottom surface node.

  » Expand the Physics Conditions node

  » Click on Thermal Specification

  » In the Properties pane, change Adiabatic to Convection. After you select Convection, a new node, Physics Values appears under the heated_plate.bottom surface node in the simulation tree.

  » Expand the Physics Values node.

  » Click on the Ambient Temperature node. In the Properties pane, type "20 C" in the Value field. Note that StarCCM+ knows how to work with units. Although the initial value for the ambient temperature was 300K (26.9C), we specify the new value as 20 C.

  » Click on the Heat Transfer Coefficient node in the simulation tree. In the Properties pane, type "1.0e4" in the Value field. Keep the "W/m^2-K" units

- Under the Boundaries node of the simulation tree, expand the heated_plate.top surface node and confirm that the Thermal Specification is Adiabatic in the Properties pane.

- Under the Boundaries node of the simulation tree, expand the heated_plate.heater surface node

  » Expand Physics Conditions node in the simulation tree and click on the Thermal Specification node.

  » Change Adiabatic to Heat Source in the Properties pane

  » Expand the Physics Values node in the simulation tree.

  » Select the Heat Source node, and in the properties pane, enter 20W for the Value.

- Save the model.
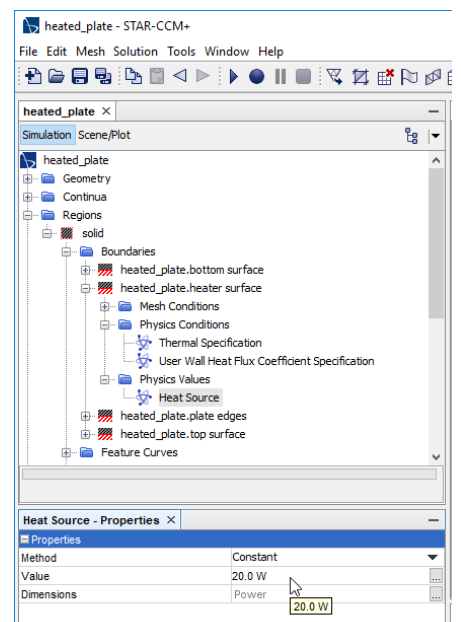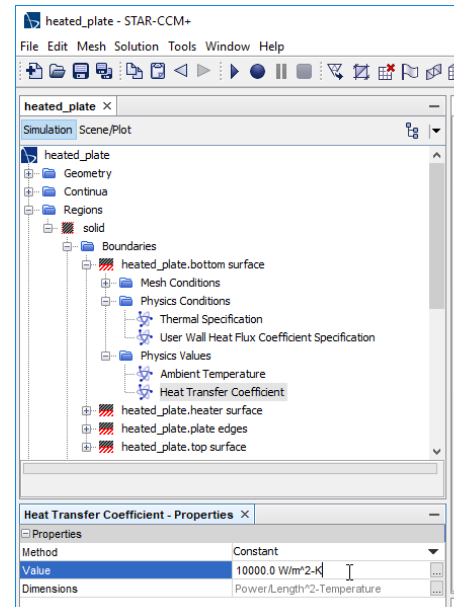
Set up the Mesh Pipeline and Create the Mesh

The parts-based meshing pipeline defines how the mesh is created. For simulations with fluid inflow and outflow, the boundary conditions will interact with the mesh creation. Therefore, as we did in the preceding steps, you need to set up the regions and boundary conditions before completing mesh creation.

Set up the Mesh Pipeline

- Under the Geometry node, right-click the Operations node and select New → Mesh → Automated Mesh.

- In the Create Automated Mesh Operation modal pane

    » At the top left of the pane, check the small box to the left of heated_plate.

    » In the bottom half of the pane, make three selections: Surface remesher, Polyhedral Mesher, and Thin Mesher .

    » Click OK.

- Expand the newly created Automated Mesh node, and expand the Default Controls sub-node.

    » Select Base Size, and in the Properties panel change "1.0 m" to "1.0 cm". Note that when you change values in the Properties pane, there is no "OK" or "Close" button. Simply type in the new value and press "return" or "enter" on your keyboard.

    » Click on the Number of Thin Layers node. In the Properties pane Number of Layers value is 2, which is fine for now. Leave this value unchanged.

Create the Mesh and Visualize It

- Right-click on the Automated Mesh node and select Execute. Mesh creation will take a few seconds and result in a lot of text being displayed in the Output panel. Mesh creation will not work if you haven't first assigned the part to a Region

- Create a Mesh Scene

    » Right click on scenes, select New Scene → Mesh

    » Use the mouse to rotate the model and inspect the mesh

    » For future reference: You can store an image of the mesh by right-clicking in the mesh scene graphics window and selecting Hardcopy …. See Figure 4 on the next page.
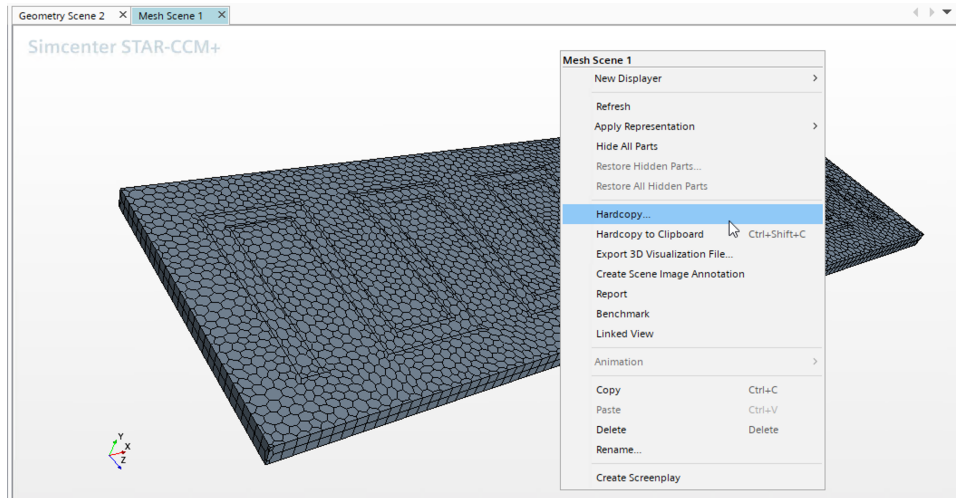
- Save the model!

Figure 4.   Mesh Scene and selecting the option to create a
hard copy of the image.

Run the model

- Set the limit on the number of iterations

  » In the simulation tree, expand the Stopping
  Criteria node

  » Click on Maximum Steps

  » In the Properties pane, set the Maximum Steps
  to 50. You can extend the solution later by
  increasing the value in the Maximum Steps
  field. Doing so allows the simulation to continue
  without losing any work.
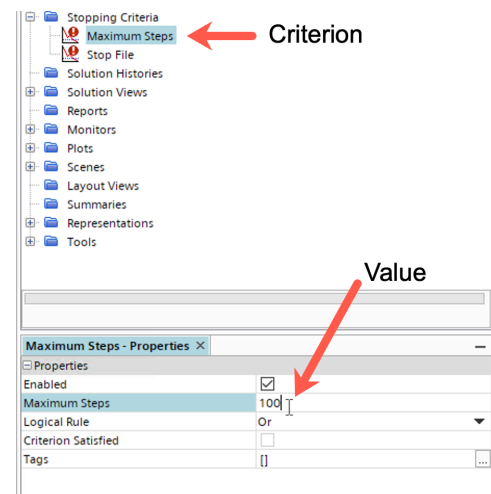
- In the ribbon at the top of the StarCCM+ window,
  click on the run icon
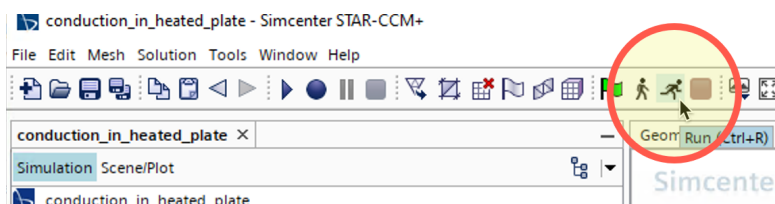
- As the solution progresses (quickly!), a new Residuals window appears showing that the
  residuals are reduced over 6 orders of magnitude in less than 50 iterations of the solver.
  Savor this. You will not achieve such an easy solution for flow problems!

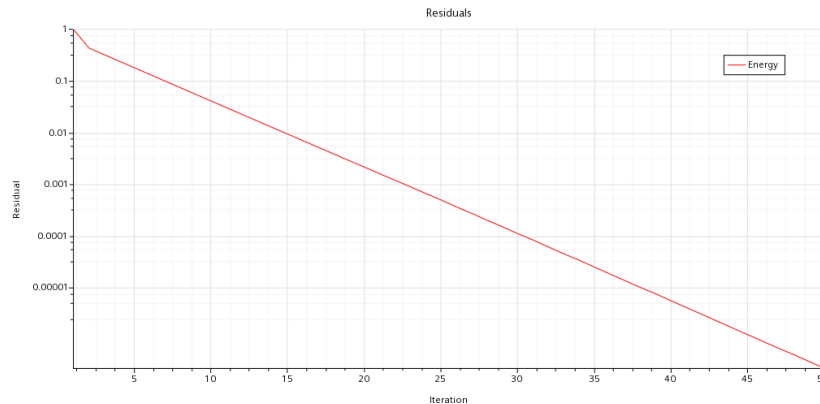- You can export the residual plot by right-clicking on the plot and selecting Hardcopy

Figure 5   Residual plot for the base case simulation.

Visualize the solution

- Create a scene to visualize the temperature on the surfaces of the plate

  » Right-click on the Scenes node in the simulation tree. Select New Scene → Scalar. Note that when working with scenes it can be convenient to toggle the Scene/Plot button at the top of the panel that holds the simulation tree. You can click on Simulation to return to a display of the full simulation tree.

  » Right click on Scalar Scene 1 and rename it to Surface temperature.

  » Expand the Displayers node; expand the Outline 1 node by clicking the + sign.

  » Expand the Parts node; expand the solid node and observe that the four surfaces of the region appear. Recall that we named those surfaces when the Region was created. If you select any of the surfaces, the Visible option in the Properties pane has a checkbox that allows you to show/hide the surface. There is no need to change anything now.

  » NOTE: The Outline 1 displayer only shows you the geometric parts! You can add more geometry displayers for complex models.

- Expand the Scalar 1 node by clicking the + sign.

  » Click on Scalar Field and in the Properties pane, click on <Select Function> in the Function value box.
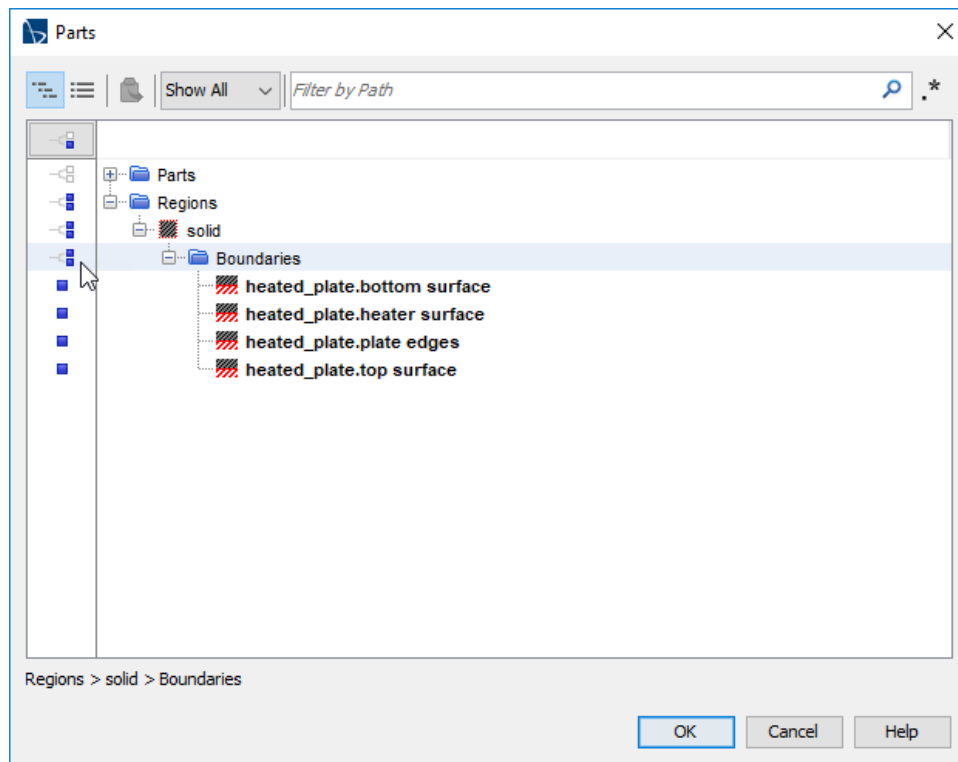
  » Scroll down to select Temperature and click OK.
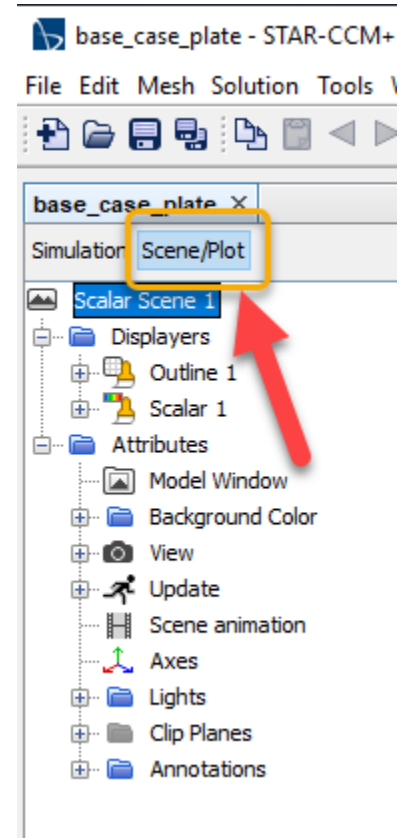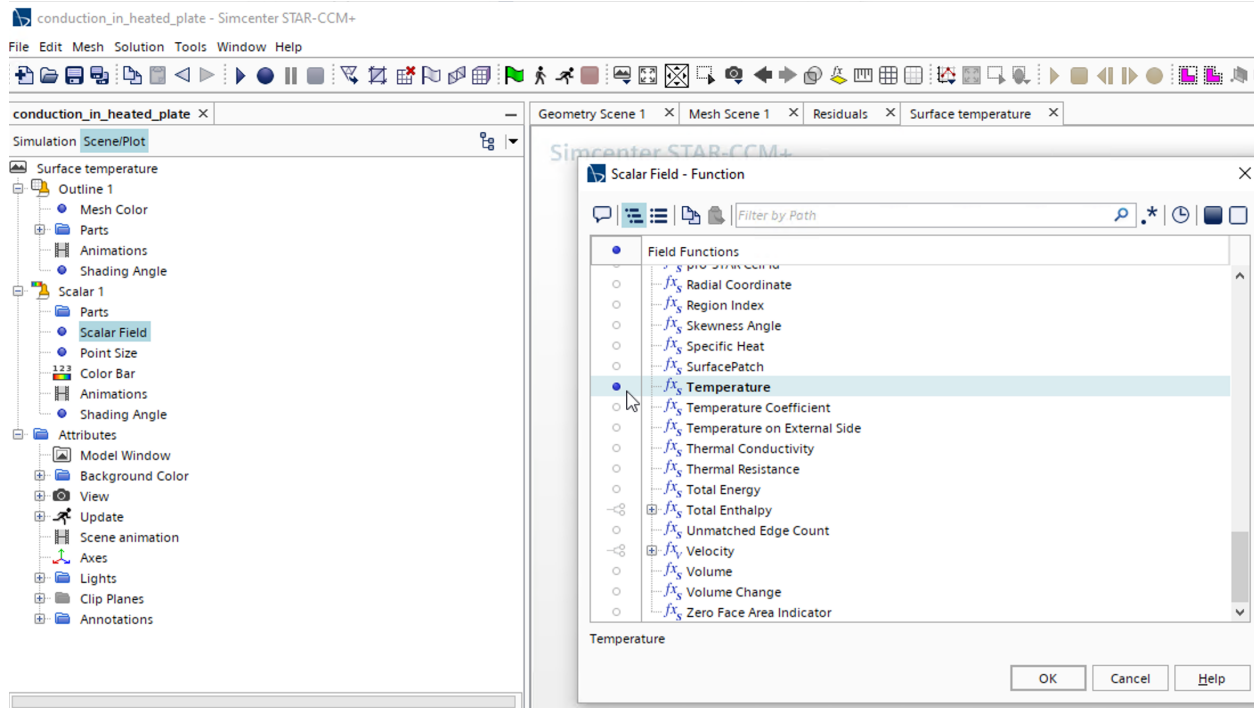
- Rotate the model in the **Surface Temperature** viewer: Nothing!
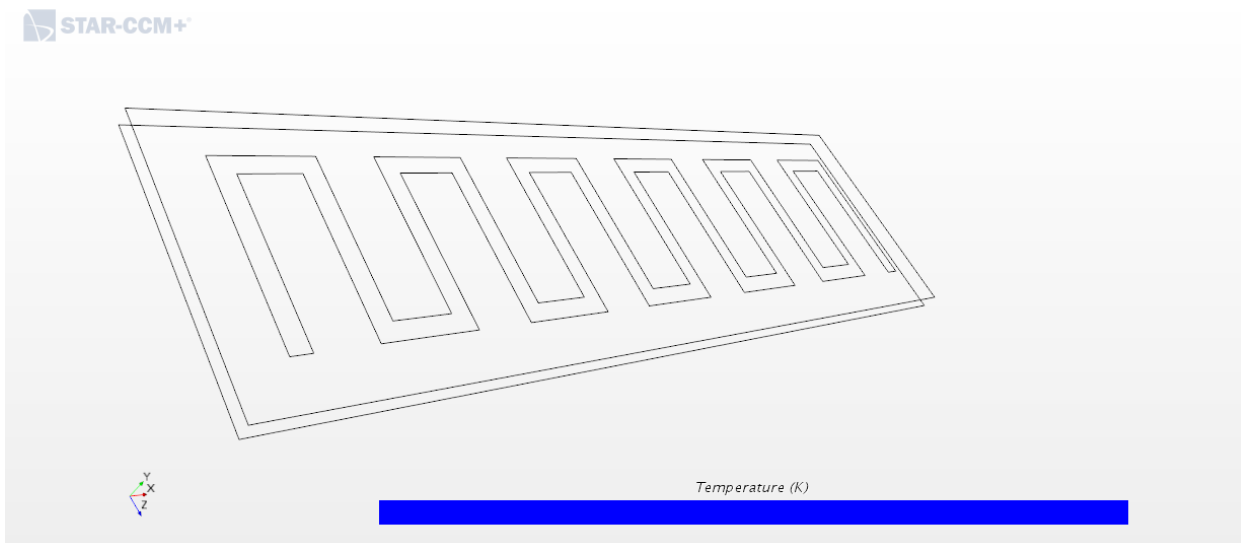
- Right-click on the Parts node under Scalar 1. Select Edit

  » Expand the Regions Node

  » Expand the solid Node

  » Select the all the sub-nodes in the solid by clicking on the tiny double blue boxes to the left of the "solid" node
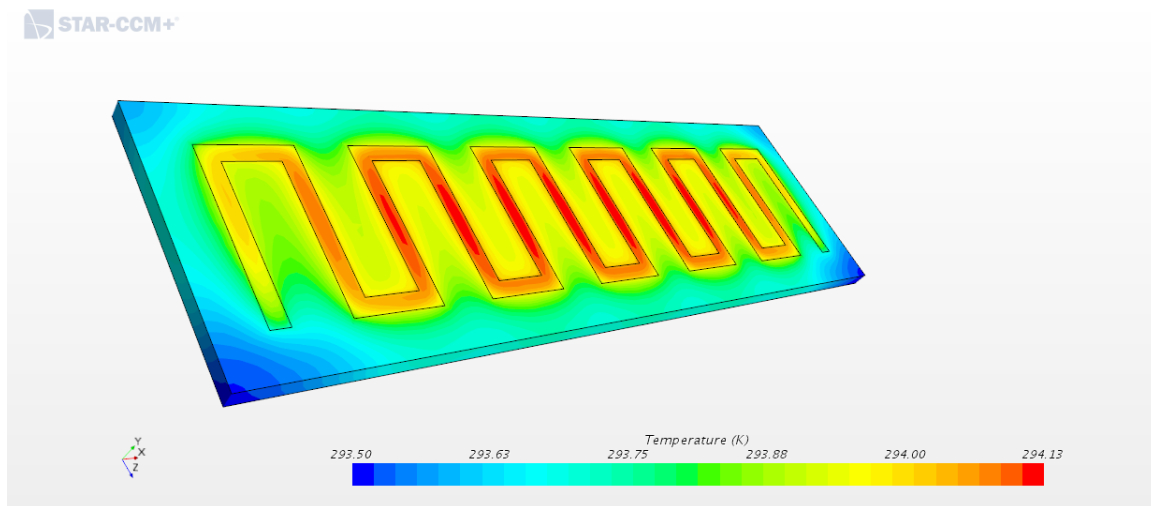
  » Click OK

- The preceding steps should show a display like the following
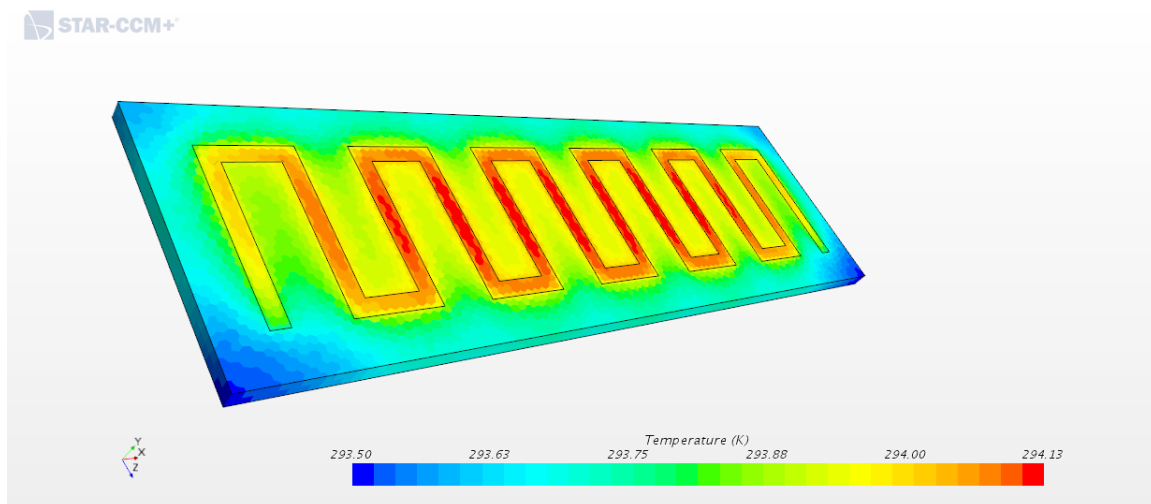
Change the appearance of the scalar display

The default is to display the temperature (or other scalar) as the cell-wise values. In the solver, the temperature is assumed to be uniform over each cell. You can improve the *appearance* of the result by displaying the temperature field (or other scalars) as values interpolated between cell centers

- Click on the Scalar 1 node.

  » In the Properties pane, click on the Automatic value next to the Contour Style label.

 » Select Smooth filled to get the following display. You can experiment with other representations.

- Change the temperature units to C

  » Click on the Scalar Field sub-node under Scalar 1, Parts.

  » In the Properties pane, select C for units

- Change the range of the color bar

  » With Scalar Field selected (from previous bullet), click in the pop-up menu next to the Auto Range option in the Properties pane. You can set the Auto Range to Off, and then manually enter Min and Max values for the range of scalar values to display. The advantage of manually setting Min and Max values is that you can choose nice round numbers for images in a report. It is probably best to leave the Auto Range option set to its default, "Min and Max Values".

- Rotate the part in the Surface Temperature Displayer to show the bottom, convectively cooled, side. Use the Save-Restore-Select-View menu (click on the camera icon) to adjust the orientation of the model. The following image shows the selection of the -Y view with +Z axis oriented upward.

  » You can also use the Store Current View option of the Save-Restore-Select-View menu to save a model orientation. That allows you to create a series of images, say displays of different field values, from the same perspective.
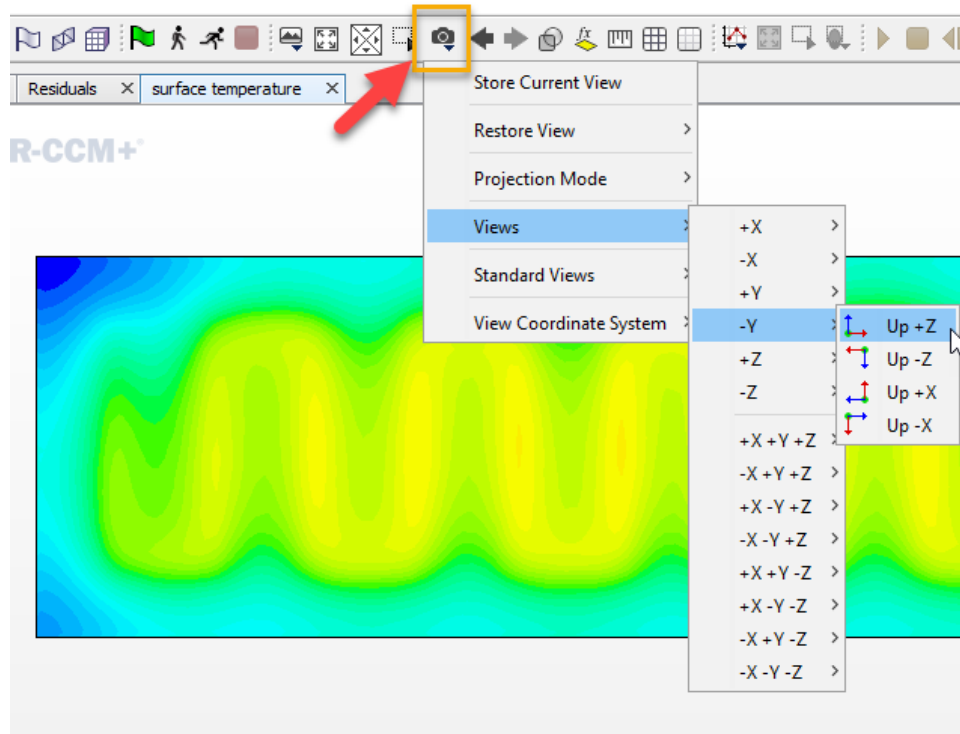
Generate Reports to Summarize the Solution

Heat Transfer Report (and energy balance)

- If you are using the Scene/Plot view of the simulation tree, click on the Simulation button to view the full simulation tree.

- Right-click on the Reports node and select New Report → Flow/Energy→ Heat transfer

- Click on the (newly created) Heat Transfer 1 sub-node

- In the Properties pane, select empty box to the right of Parts and select all of the Region surfaces (not part surfaces), Or click on the box with the […] in it
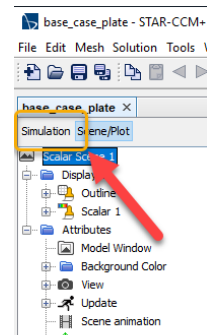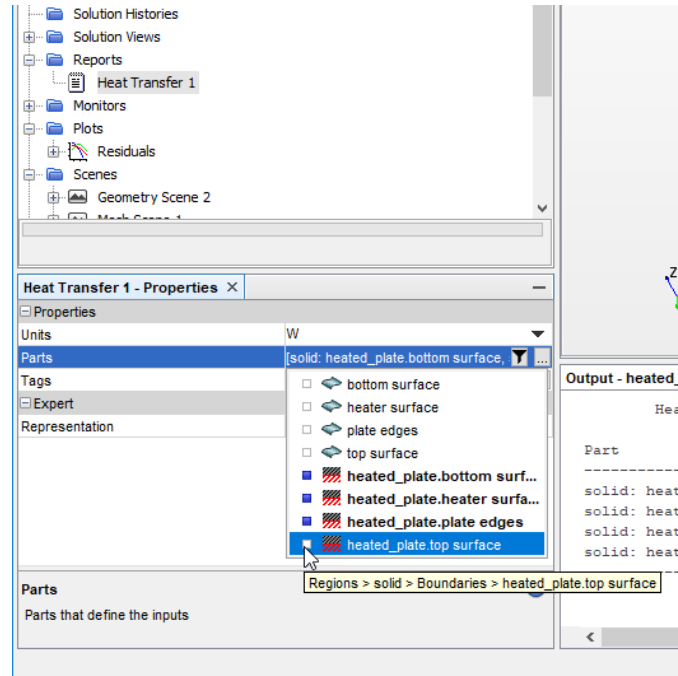
- Right click on the Heat Transfer 1 node under reports and select Run Report. The results are displayed in the Output window. The report shows that the heat input from the serpentine heater is –20W and +20W from the convectively cooled surface. Positive heat transfer is out of the domain. The total is nearly zero, which indicates that the overall energy flow by heat transfer balances.

Rerun the Solution with different parameters

Here are some ideas for exercising the model.

- Increase the heat load

- Change the convective coefficient

- Change the material

- Refine the mesh

- …?

Create Derived Parts for Additional Visualization

To visualize the solution or the mesh on plane sections through the domain, we need to create those sections as Derived Parts. Other types of derived parts are points, lines and streamlines. Points and lines are useful for extracting values at specific locations within the domain or on surfaces – think of a velocity profile at an exit plane. Streamlines are a special kind of derived part that uses the solution (the velocity field) to render a geometric object (the streamline).

Plane Section for Internal Temperature or Mesh

Create a plane section to view the temperature distribution inside the plate. There are several ways to create and then locate the plane. We will create the plane in the current scalar scene.

- Right-click on Derived Parts in the simulation tree. Select New Part → Section → Plane. The result is the following image in the graphics pane
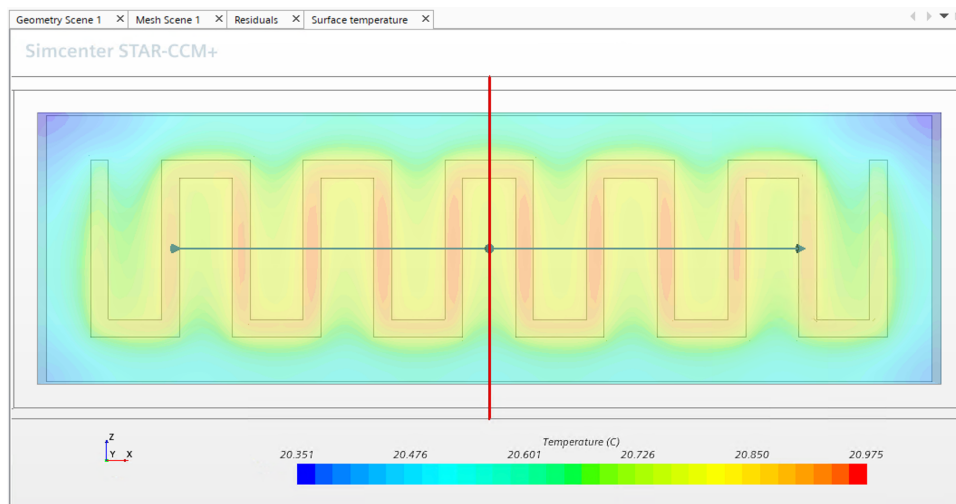

- Drag the arrow to change the orientation of the plane. Drag the outline of the plane to change the location of the plane. While it is possible to locate the plane with mouse gestures, it is easier to get a precise orientation and specifying numerical coordinates for the origin and normal direction for the plane.

- In the Create Plan Section panel to the left of the graphics window, enter the following coordinates for the origin and normal of the plane

| Origin	Normal
5.0 cm	0.0 m
0.0 cm	0.0 m
−1.5 cm	1.0 m

- Click Create
- Click Close

- Create a new scalar scene to visualize the derived part. You can do this with a copy/paste operation on the existing (temperature) scalar scene.

  » Click on the Surface Temperature scene (was Scalar 1) node to select it.

  » Right-click, select Copy.

  » Click on the (higher level) Scenes node in the simulation tree, and right-click to select Paste. The result is a duplicate scene

  » Rename the new scene as "temperature plane section".

  » Expand the Displayers node and expand the Scalar 1 sub-node

  » Click on the Parts node, and in the Properties Panel, click the […] box

  » In the Parts dialog box and select the newly created plane section (under Derived Parts). The following image shows the five steps in the process.
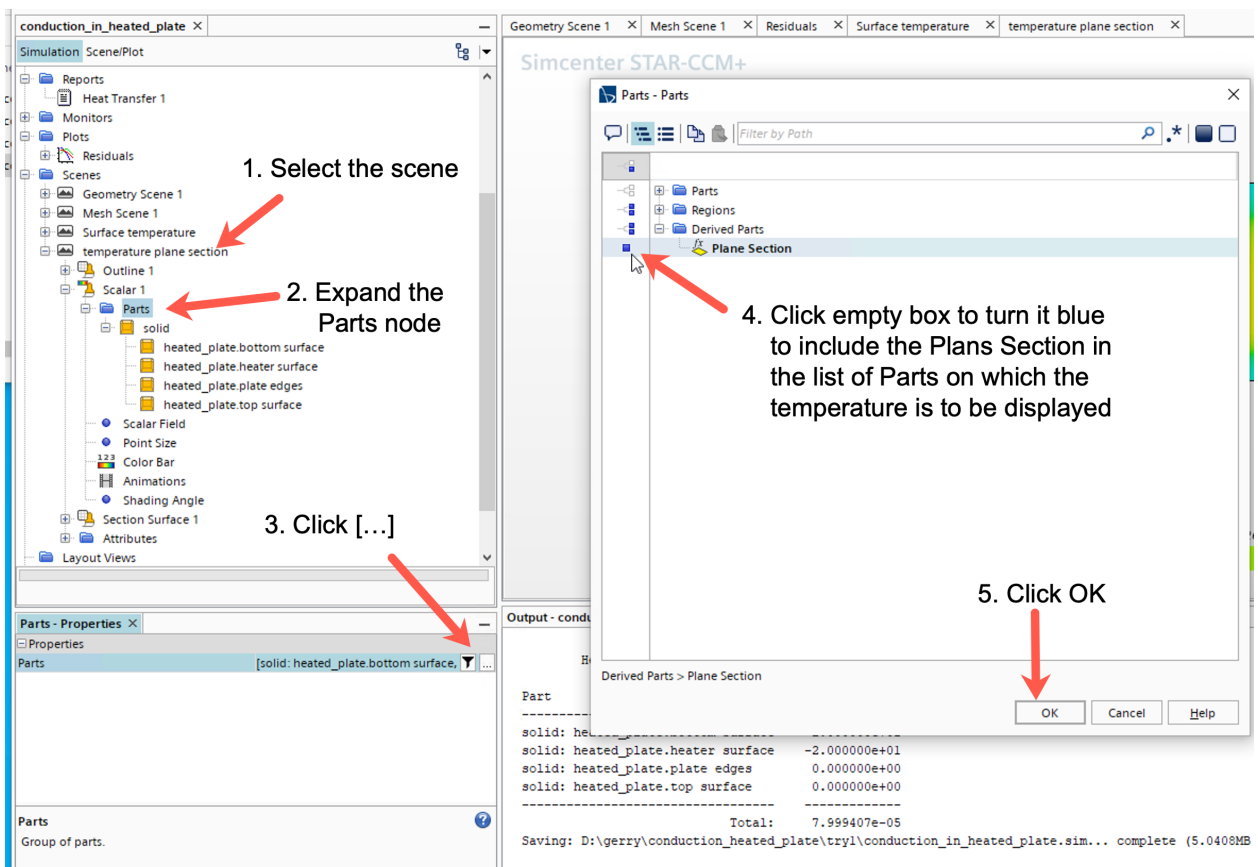
At this point, the newly created plane is obscured by the top and bottom surfaces.

- Turn off the display of temperature on surfaces of the plate

    » Under the Scalar 1 sub-node, expand the Parts node of the Scene,

    » Click on each of the parts under the solid node and uncheck the Visible box in the Properties Panel
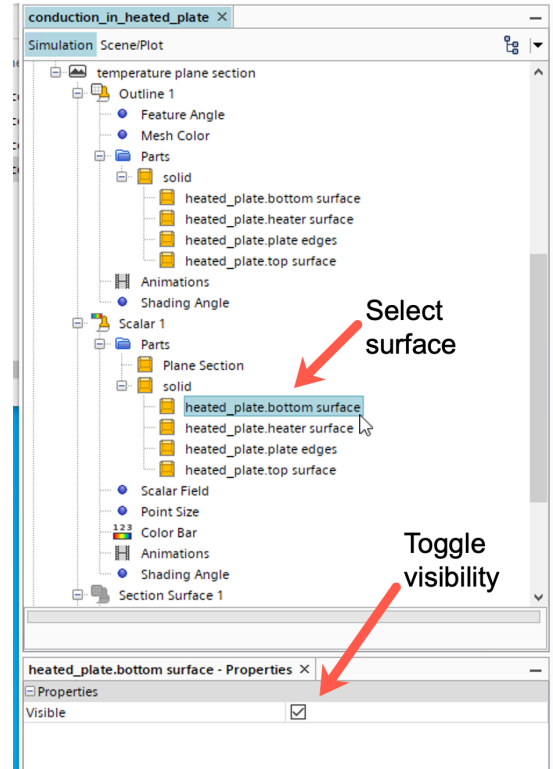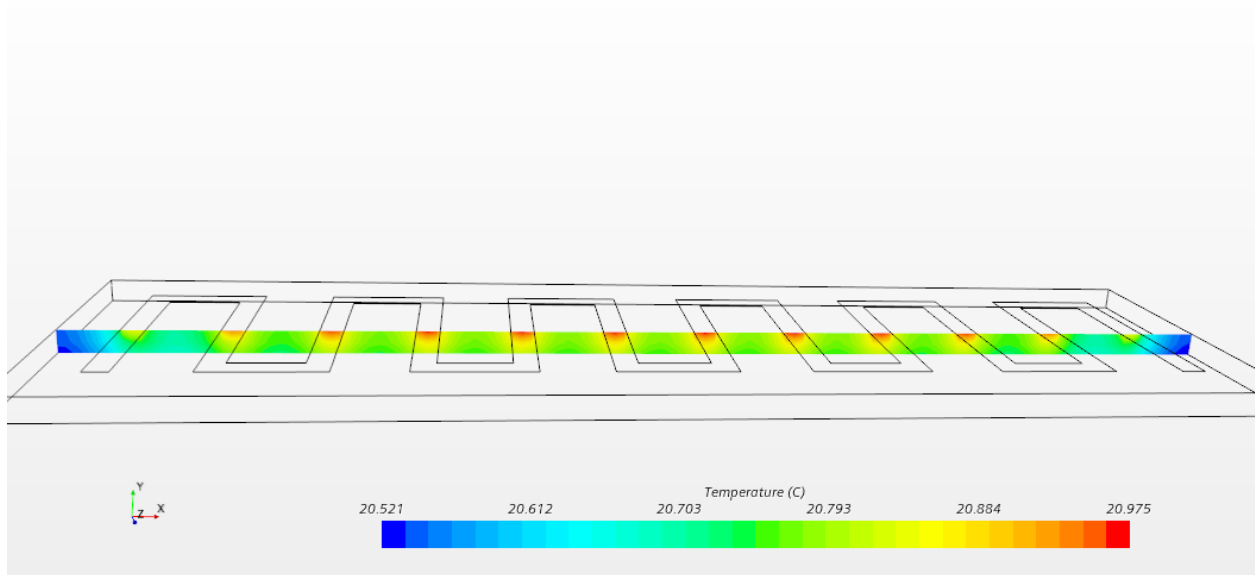

The solid geometry surfaces may be obstructing the temperature.

- Create only the outline of the geometry

    » Click on the Outline 1 node at the top of the Scene.

    » In the Properties Panel, uncheck the Surface box (if it is checked) and make sure the boxes for Outline and Feature lines are checked.

The preceding steps should produce a plot like that at the top of the next page.
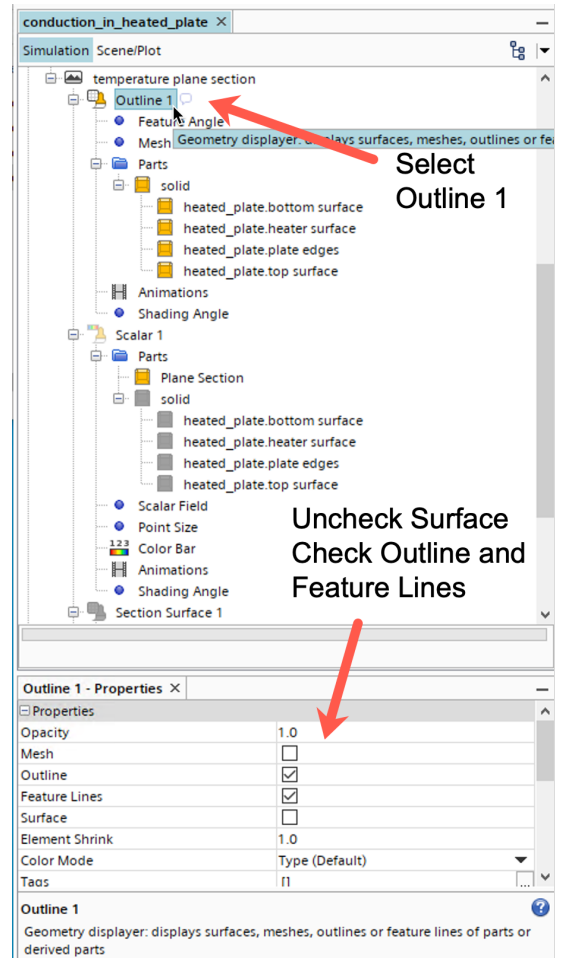
Line Probes for Temperatures along the center of the plate on top and bottom

Create a line probe

- Right-click on Derived Parts → New Part → Probe → Line …

- Adjust the end points to be the values in the table below

Point 1	Point 2
0.0 cm	10.0 cm
0.0 cm	0.0 cm
−1.5 cm	-1.5 cm

- Change the resolution to 80

- Click Create

- Click OK

- Expand the Derived Parts node and make a copy of the Line Probe

  » Select "Line Probe" and right-click Copy

  » Click on the Derived Parts node and right-click Paste

- Change the end point values for Y in the new line probe by directly editing in the Properties Pane

Point 1	Point 2
0.0 cm	10.0 cm
2.0 mm	2.0 mm
−1.5 cm	-1.5 cm

- Rename the line probes to "Centerline on top" and "Centerline on bottom"

Create and XY plot of temperatures along the centerline

The preceding steps have created two new line probes.

- Right-click on the Plots node of the simulation tree. Select New Plot → XY Plot

- Expand the X Type node.

- Click on the Vector Quantity sub-node. Note that the Value is [1.0, 0.0, 0.0] m, which means that the horizontal axis will be the x-direction. Leave that value

- Click on the Y Types node to expand it. Expand the Y Type 1 node

  » Click on the Scalar Function sub-node

  » In the Properties Panel, click on <Select Function>

  » Scroll through the list of Field Functions and select Temperature

  » In the Properties Panel, change the units from K to C

- Click on the XY Plot 1 node at the top of the newly created Plot

  » In the Properties Panel, click on […] for the Parts value

  » Select the two Centerline Line probes

  » In the Properties Panel, change the title to "Temperature along centerlines"

- Adjust the line type of the data

  » Expand the Centerline on Bottom node (newly added to the plot)

  » Click on the Line Style subnode and change the Style to solid

  » Click on the Centerline on Bottom node again and Check the Sort Plot Data box (you may need to scroll

The preceding operations should produce a plot like that at the top of the next page.

- Save the Plot

  » Right-click on the graphics window and select Hardcopy from the pop-up menu

Temperature along centerlines
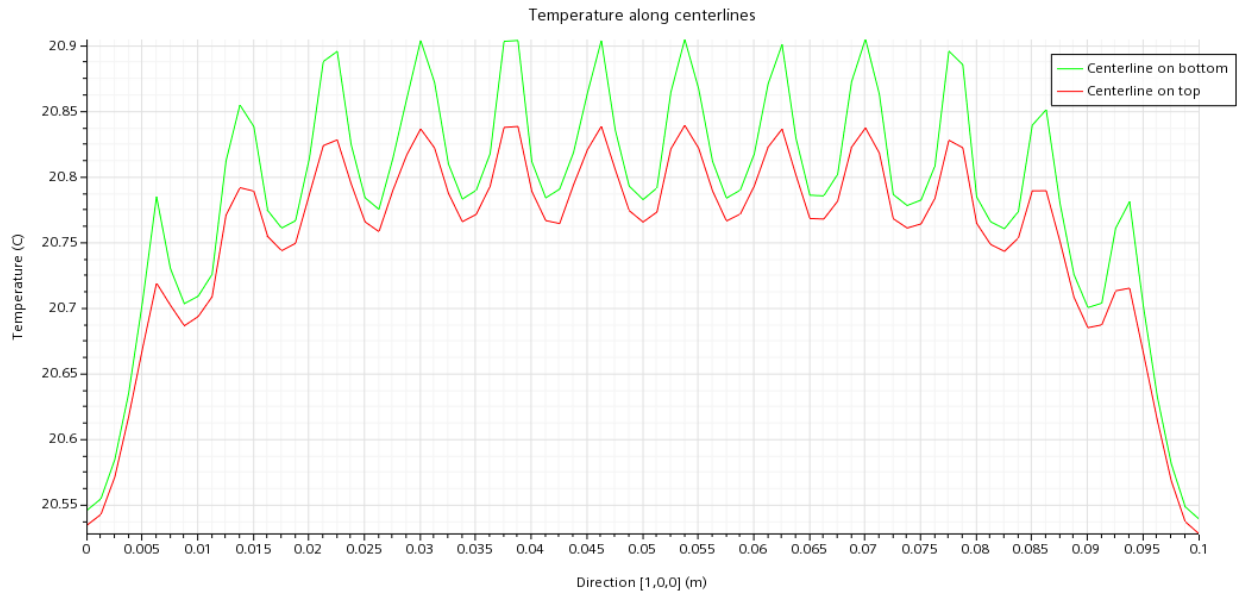
Refine the mesh on the heater surface

Global mesh parameters are set via the base size property. In many CFD simulations, local mesh refinement is beneficial, and there are several strategies for controlling the local mesh. We will show a basic approach with surface mesh control.

In general, the mesh should be finer in regions where the gradients of important quantities are steepest. For the heated plate problem, temperature the most obvious quantity of interest, although we could also focus on the variation of heat flux on the convectively cooled surface. Looking at the surface temperature and midline plots from the base case, the steepest temperature gradients are on the face with the serpentine heater strip. We take this as a cue that refining the mesh should focus on the heater surface.

Note that surface mesh properties can be set with the *Reference Values* of the solid mesh via sub-nodes of the mesh continua node in the simulation tree. It is instructive to inspect these values, but not change them.

Before proceeding, it would be good to save the current simulation file, and then do a Save As operation to create a new simulation file. For example, the existing model could be called "heated_plate_base_case" and the new model could be called "heated_plate_medium_mesh". It is probably a good idea to put each of these models in their own folder of the file system. That way you can save exported data, plots and other images in folders with the related simulation file.

Set local mesh properties

Instead of controlling the mesh with global properties of the *Solid Mesh Continuum*, we will use the local mesh properties of the *Boundaries* under the *Regions* node.

- Expand the *solid* node under *Regions* in the simulation tree

- Expand the *Boundaries* node and then expand the "heated_plate.heater surface" sub-node

- Expand the *Mesh Conditions* node

  » Click on the *Custom Surface Size* sub-node.

  » In the Properties panel, check the box of *Custom Surface Size*. This is important. If this option is not checked, change the mesh size parameters in the following steps will not have any effect when the mesh is generated. Although it is a potential trap in the user interface, being able to turn the *Custom Surface Size* feature on and off is a benefit because it allows you to toggle local mesh refinement on an off without deleting all of the local mesh refinement parameters.

- Expand the *Mesh Values* node under the "heated_plate.heater_surface" node

  » Click on the "Surface Size" sub-node and notice the two options in the Properties pane: *Relative/Absolute* and *Size Method*.

  » In the Properties Pane, change the *Relative/Absolute* property from "Relative to base" to "Absolute". Note that "Relative to base" will work, but that means any change in the base size will also change the surface mesh size. You may or may not want to use that strategy in your mesh refinement. I prefer the absolute control method so that I can change one aspect of the mesh refinement at time. The disadvantage is that you

may forget to change all of the mesh parameters during a mesh refinement study. Using the "Relative to base" option is certainly simpler.

> » Leave the *Size Method* property at its default, "Min and Target"

- Expand the *Surface Size* subnode of *Mesh Values*

  > » Click on *Absolute Minimum Size* and change the value to 0.2 mm. This parameter sets a lower limit on the size of mesh features on this surface. Also note that this parameter is only available because we are using the "Absolute" form of specifying the mesh.

  > » Click on *Absolute Target Size* and change the value to 0.5 mm.

Generate the new mesh

Before generating a new mesh, you will need to delete the existing mesh. That will also clear the simulation data from the previous analysis. For this simple model, it is easy (and quick) to generate a new mesh and a new solution. For more complex models, you will want to carefully consider how to save results before proceeding with a new mesh.

- Click the Clear Mesh icon at the top of the StarCCM+ window

- Click the Generate Mesh icon

- When the mesh generation is complete, rerun the Element Count report. The following text should appear in the Output pane

```
      Element Count

   Part                          Element Count
   -----------------------------------------
   solid                          1.255100e+04 cells
   heated_plate.bottom surface    2.450000e+03 faces
   heated_plate.heater surface    3.497000e+03 faces
   heated_plate.plate edges       2.400000e+02 faces
   heated_plate.top surface       5.496000e+03 faces
                                  -------------
                     Total:       2.423400e+04 element
```

- Open the mesh Scene and inspect the surface mesh. The following images show the surface mesh on the heater-surface side and the convection-surface side. Because the surface mesh control was applied to the heater surface, the elements on that surface are smaller. StarCCM+ creates a smooth variation in cell size from the specified surface properties, expanding outward toward the base size of the global mesh.
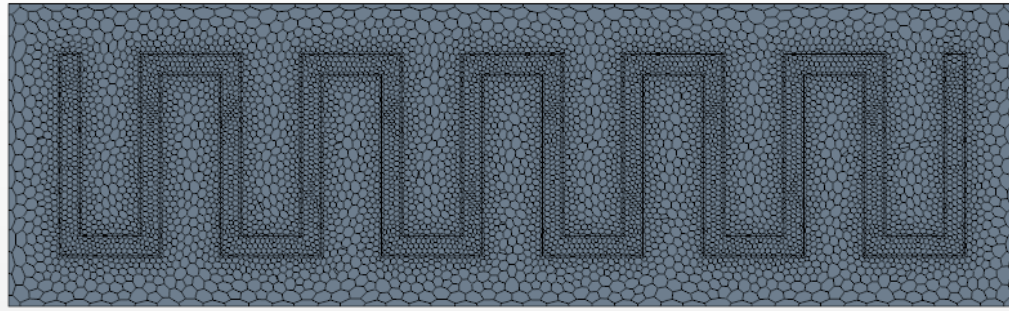
Mesh on the heater surface
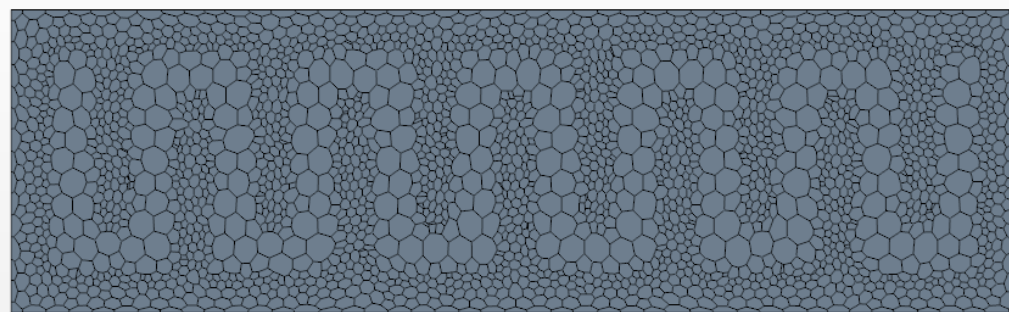
Mesh on the convective surface

Run the simulation on the new mesh

While it is possible to restart a simulation with information from a coarse mesh solution, the model of heat conduction is simple enough that we can easily (and conveniently) start from scratch. Deleting the prior mesh has already cleared the previous simulation results. However, to start with a new simulation, we can also click select "Clear Solution" from the *Solution* menu, and then confirm by clicking "OK" to the dialog box.
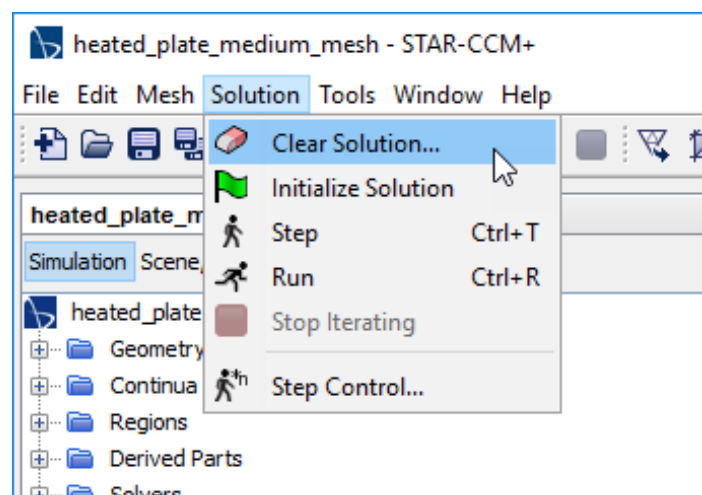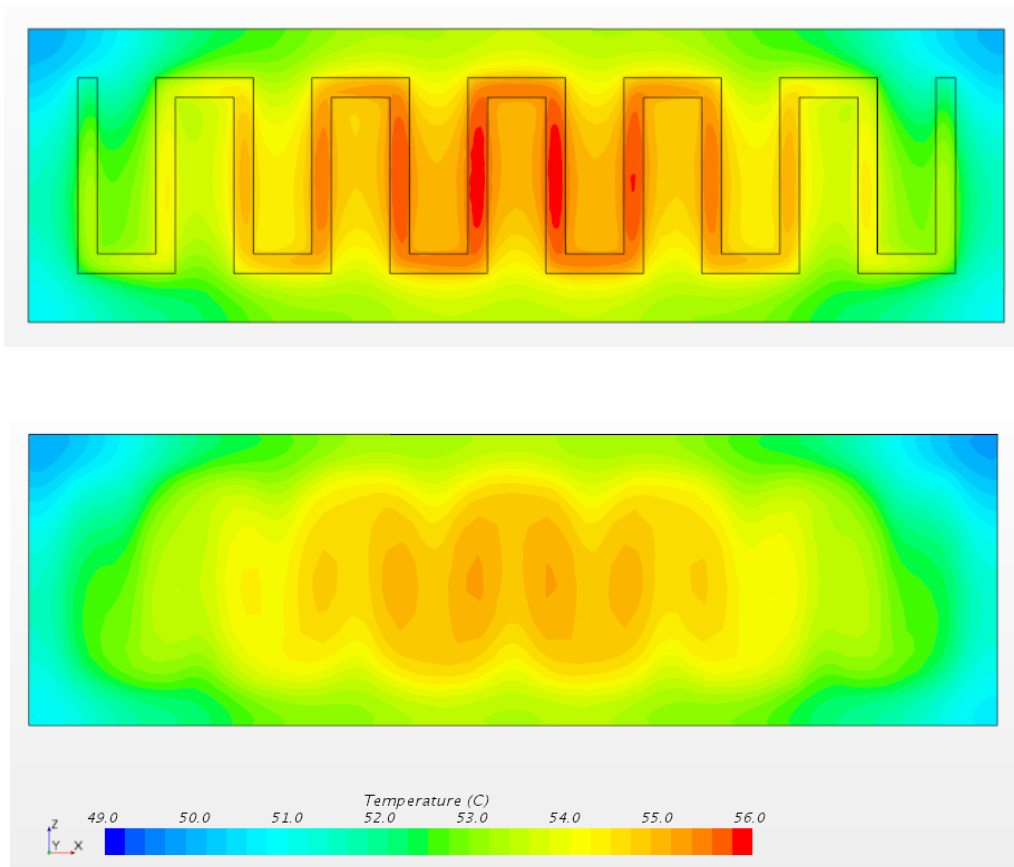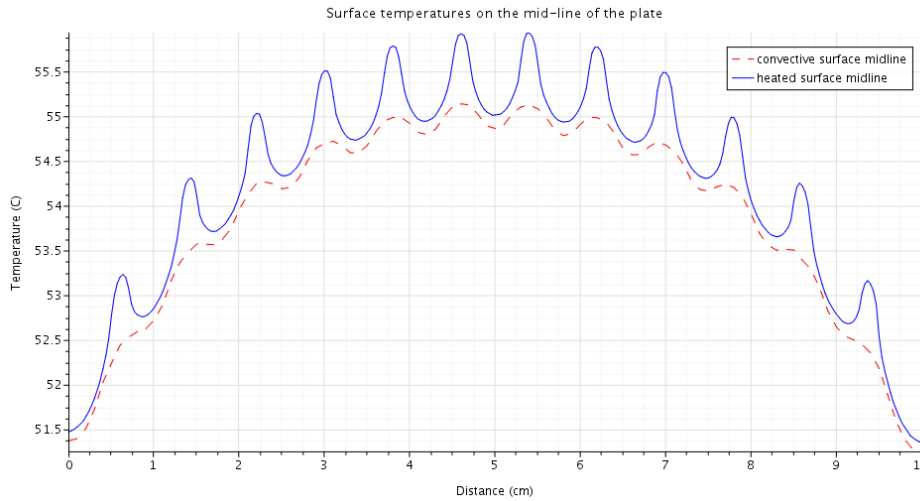
Run the solution

- Click the run icon
- Note that the solution takes more iterations than the base case with a coarser mesh.

Medium Mesh Results

There is little qualitative difference in the surface temperature distributions

Surface temperatures on the mid-line of the plate

Since the energy balance holds regardless of the mesh size, the results of running the heat transfer report are essentially the same. However, since the larger model (finer mesh) takes longer to converge, the energy balance is not as close to zero after 500 iterations as it was in the base case after 220 iterations.

```
        Heat Transfer of Boundary Heat Transfer on Volume Mesh

Part                                     Value (W)
--------------------------------      -------------
solid: heated_plate.bottom surface      9.999999e+01
solid: heated_plate.heater surface     -1.000000e+02
solid: heated_plate.plate edges         0.000000e+00
--------------------------------      -------------
                           Total:      -1.170449e-05
```